TUTORIAL 2: FRAME ANALYSIS

Concepts

Finite element analysis; frame structures; meshing; beam sections. Post processing.

Problem Statement

A two dimensional frame analysis is performed. Post processing results relevant to frame analysis are generated.
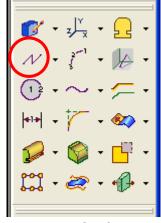
Steps in a Finite Element Analysis

1. Build model: Solid, shell, plate or wireframe
2. Apply boundary conditions
3. Mesh model: Create elements and nodes, assign material and structural properties
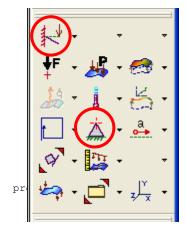4. Solve model
5. View and post process results

Model Construction

1. Start a new model file using a descriptive name.
2. Make sure that I-DEAS is in the *Design* application and the *Master Modeler* task.
3. Choose Customary Units for the model under the <u>Options </u>drop down menu: *Foot (pound f)*.
4. Place points at the locations of the frame's three joints, A(0,0), B(20,0), C(30,10). The points can be generated using the *Points* command located under the *Polylines* icon.
5. Alternatively you can use the Lines command under the *Polyline* icon in combination with *Options…* under RMB.]
6. Join the points together to form the frame using one of the line commands.
7. Name the structure *Frame* using the *Name Parts…* function under the *Manage Bins…* (bottom center of the middle icon panel).
8. Save the model file.

Application of BCs and Loads

1. Move to the *Simulation* application and the *Boundary Conditions* task.
2. If the top left icon is not selected to be *Linear Statics* do so now.
3. Select the *Displacement Restraint…* icon in the center of the top icon panel.
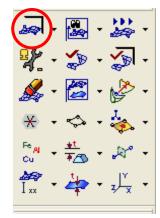4. Select the *Displacement Restraint…* icon.

5. I-DEAS now prompts you with the *FE Model Create* form.
   a) Think of the FE Model as a basket to hold all of the finite element items you will create. Similarly, the solid model stored by I-DEAS is a basket of all the geometric information about your part or structure.
6. Create an FE model with the name *Frame FEM*.
7. Use the *Displacement Restraint…* icon and the *RMB, Location on Wireframe* to clamp the bottom left joint (A) on the frame.
8. Put a roller on the bottom right joint (B) using the *Displacement Constraint…* icon.
9. Use the *Force…* icon (directly below the *Linear Statics* icon) to apply a 1.175e6-lb downward force on the top joint.
10. Save model file.

Define the Frame Members Cross Section

1. Move to the *Beam Sections* task.
2. Choose *Circular Beam* under the *Solid Rectangular Beam* icon (top right of top icon panel).
3. Enter the Outside Diameter (12-in) and the Inside Diameter (6-in) in the I-DEAS Prompt window. Remember that the model is defined in feet.
4. Accept the cross section (MMB).
5. Put the cross section away using the *Store Section* icon in the top menu panel.
6. Name the cross section *Frame Cross Section*.

Mesh the Model

1. Move to the *Meshing* task.
2. Choose the *Define Beam Mesh…* icon under the *Define Shell Mesh…* icon located in the top left corner of the top icon panel.
3. Choose GENERIC_ISOTROPIC_STEEL to be your *Part Material*.
4. Shift-click to select all of the frame members.
5. On the *Define Mesh* form that appears select *Beam Options…*
6. Push the *?* next to the top input box and select *Frame Cross Section*. Hit OK.
7. Now select the *Googly Eyes* on the *Define Mesh* form.
8. This starts the *Modify Mesh Preview* tool. This tool allows you to control the meshing process and see what the mesh will look like before it is generated.
9. The mesh nodes to be constructed on the model are now shown as blue stars.

10. To add more nodes (and elements) cancel the *Modify Mesh Preview* form. Now change the *Element Length* to 1 on the *Define Mesh* form.
11. Choose the *Googly Eyes* again. Notice that the density of nodes has increased. Accept this mesh by selecting *Keep Mesh*.
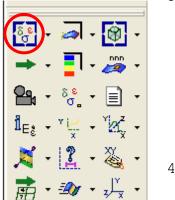12. The solid model and finite element model are now complete. Save the model file.

Model Solution

1. Move to the *Model Solution* task.
2. Generate a Solution Set by selecting the *Solution Set…* icon located at the top center of the top icon panel. Select the *Create…* button and name the solution set *Frame Solution* in the *Solution Set* form.
   a) A solution set is a basket that stores and organizes all of the information generated by the solution. It is analogous to the FE Model and the Solid Model.
3. Select the *Output Selection…* button. Highlight *Element Forces* in the scroll menu and choose *Store* in the drop down menu under the *Store/List* heading. Click *OK* then *Dismiss*.
4. Now solve the FE model by selecting the *Solve* icon located under the *Manage Solve* icon.
5. Check the I-DEAS *List Window* to see that no error or warning occurred during the solve.

Post Processing

Displacements

1. Move to the *Post Processing* task.
2. Choose the *Results…* icon to choose which results to display.
3. The *Results Selection* form is now displayed. The large window shows all of the results available for display. The two smaller windows on the right, *Display Results* and *Deformation Results*, show what information is to be displayed. The upper window is typically used to display stress, reactions, etc… The lower window displays deflection results.
4. To display deflection alone *Clear* the *Display Results* with the clear button.
5. Now display the results using the Display icon located directly below the *Results…* icon.
6. Specific displacements can be determined by using the *Probe* icon.

7. After selection Probe simply click on any portion of the structure to determine the deflection at that point. To view x and y displacements separately go back to the *Results…* form and change the *Component* value under the *Deformation Results* header.

Stresses: von Mises

1. On the *Results Selection* form select the stress entry from the scroll list and load it into the *Display Results* field using the triangle. Click *OK.*
2. Display the results.
   a) If an error requires that "*fast" contour display* be turned off select the *Display Template…* icon locate top center of the top icon bar. Choose *Contour…* and deselect the *Fast Display* option.
3. Use the Probe icon to sample the stress at several locations.

Section Stresses

1. Select Redisp*lay with Autoscale* to refresh element and node symbols.
2. Select *element forces* for the *Display Results…* using the *Results…* icon.
3. Select the *Beam Post Processing* icon. On the menus that appear choose *Contour On X Section*, select *Beam*. Choose an element on the frame. Select *Position Along* Beam, End of Max Stress. Select Data Component, Axial Normal. Select *Shaded Image*, *Execute*.

Bending Moments

1. Select the *Beam Post Processing* icon. On the menu choose and select *Force & Stress, Data Component, Force, SRSS Bending Moment*; *Projection Plane, Screen Plane*; F*orce & Stress, Execute*.